International Journal Of Engineering Research And Development
e- ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 20, Issue 11 (November, 2024), PP 1441-1449

Injection Molding of Polymer Test Specimens on In-house Molding Device

Danijela Pezer, Jurica Atelj
University Department of Professional Studies, University of Split, Split, Croatia
Corresponding Author: Danijela Pezer

ABSTRACT
The paper presents the process of manufacturing and testing a device for plastic injection molding. The device uses a drill stand and a heater with a digital controller for precise heating and injection of plastic into the mold. A two-part mold with one mold cavity, a sprue and a gate, is made of EN AW-6082 aluminum on a CNC milling machine. The mold cavity is designed for injection molding of polymer test specimens 1BA according to the ISO 527-2 standard. The injected test specimens are made of polypropylene (PP) and will be used to test mechanical properties (tensile strength) on a in-house tensile test machine. Autodesk Moldflow Adviser software was used to simulate the flow of the polymer melt in order to optimize the gate position and select optimal process parameters in order to reduce errors during the injection molding process, which affect the quality of the finished product.

Keywords: *Polypropylene, Injection Molding, Design, Mold, Cavity, Test specimens.*

Date of Submission: 14-11-2024 Date of Acceptance: 29-11-2024

I. INTRODUCTION

Injection molding (IM) of plastics is a key process in the polymer materials industry, enabling the precise and efficient production of a variety of parts and products. The injection molding process of polymer materials has found applications in various industries, from automotive to medical [1,2]. The injection molding process requires precise control of temperature, pressure and speed to ensure the quality and accuracy of the final product [3, 4]. An important component of this process is the injection molding machine, which is used to inject the plastic into the mold. The injection molding machine has a complex design that allows control over all process parameters, including temperature, pressure and injection speed. The advantages of injection molding of plastics are numerous. Primarily, it enables the rapid production of high-quality parts with tight tolerances. In addition, the IM process enables cost-effective mass production, thereby reducing production costs in the long term. In addition, injection molding of plastics allows great flexibility in product design, which is essential in today's market where innovative solutions and rapid adaptation to changing market requirements are sought. Also, this process is relatively environmentally friendly because it allows for the recycling of plastic [5] and reduces the amount of waste.

For the purpose of this research, the polymer material polypropylene (PP) was selected for the manufacture of test specimens, as it possesses high strength and stiffness, making it an ideal material for testing mechanical properties such as tensile strength.

II. MATERIAL AND METHODS

2.1 Materials

Polypropylene (PP) is one of the most widely used polymers in industry [6] due to its exceptional mechanical, chemical and thermal properties and ease of processing [7]. PP is also a flexible material with high impact resistance, which contributes to its ability to withstand various loading conditions during tensile tests. In addition to its favorable mechanical properties, PP is a lightweight material with a low density, which facilitates the manipulation of samples during the injection molding process and tensile tests. This characteristic can also have an impact on the final performance of the material in certain applications, such as the automotive industry or packaging. PP is resistant to many chemicals including most acids, alkalis and solvents, as well as aggressive environments, making it suitable for a wide range of applications and the production of various components and products. Its lower viscosity at higher temperatures facilitates the injection molding process during injection molding.

Table 1 shows the properties of PP. The values given in the table may vary depending on the additives in the material and the manufacturer. The impact resistance of polypropylene depends on the production process and additional reinforcements in the material.

Table 1. Material properties of polypropilene [8]

Material properties – polypropiylene (PP)		
Property	Unit	Value
Flexural strength	MPa	10 – 35
Tensile strength	MPa	25 – 40
Impact resistance	kJ/m²	5 - 150
Density	g/cm³	0,91 - 0,94
Softening point, T_g	°C	140 - 150
Melting point, T_m	°C	160 - 166
Water absorption, 24 hr	%	0,01

In the plastics industry, it is important to ensure that a material meets certain strength standards to ensure product safety and reliability. The test used to assess the strength of a material is the tensile strength test, and the test specimens are standardized. Figure 1 shows a 1BA test specimen according to the ISO 527-2 standard, and Table 2 gives dimensions of 1BA test specimens, that need to be prepared for injection molding of plastics. Every polymer material has a tendency to shrinkage upon cooling, for polypropylene the values are in the range of 1 to 2% and the actual dimensions of the mold are increased by this amount to obtain the most accurate dimensions of the finished product during volumetric shrinkage. It is important to ensure that the specimens are homogeneous and not subject to defects that could affect the test results. The ISO 527-2 standard [9] provides important guidelines and methodology for tensile strength testing of polymer materials, allowing manufacturers and researchers to objectively assess the performance of the material. Precisely defined sample preparation and testing procedures ensure reliable results that are essential for the development of high-quality and safe plastic products.

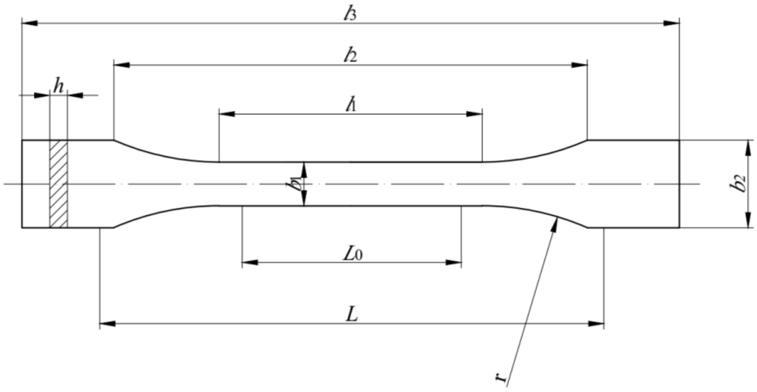


Figure 1: Test specimen type 1BA according ISO 527-2 standard

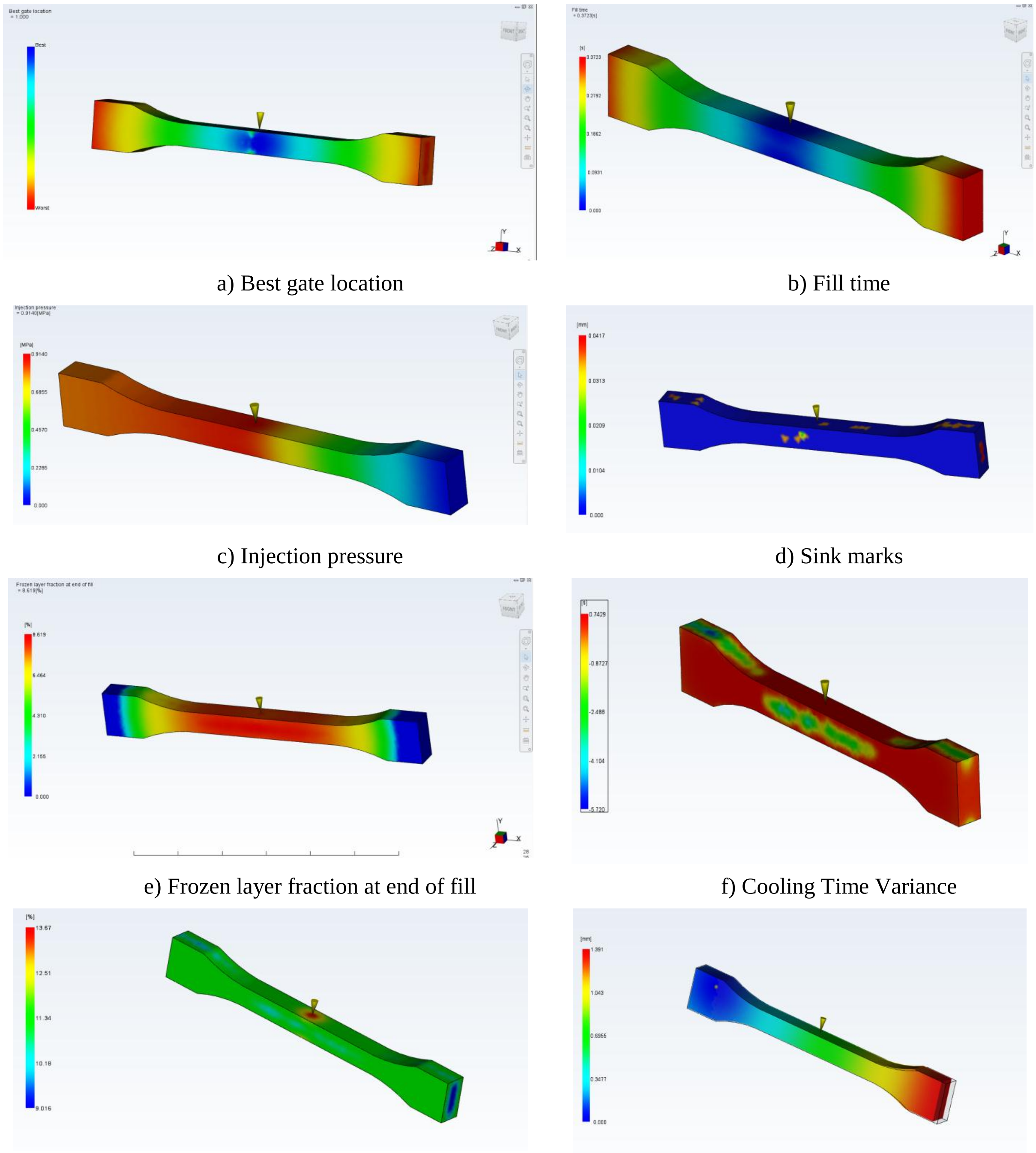
Table 2: Dimensions of type 1BA test specimen in millimetres, according to ISO 527-2 standard [9]

	Specimen type 1BA, dimension in millimetres	
l_3	Overall length	≥75
l_1	Length of narrow parallel-sided portion	30,0 ± 0,5
r	Radius	≥ 30
l_2	Distance between broad parallel-sided portions	58 ± 2
b_2	Width at ends	10,0 ± 0,5
b_1	Width at narrow portion	5,0 ± 0,5
h	Thickness	≥ 2
L_0	Gauge length	25,0 ± 0,5
	Gauge length (acceptable if required for quality control or when specified)	50,0 ± 0,5
L	Initial distance between grips	l_2^{+2}

A 3D model of the mold was designed, according to the rules of technological design of injection molded products, to simulate the flow of polymer melt in Autodesk Moldflow Adviser software. The selected mold material is polypropylene (PP).

Moldflow software is used for prototyping and helps to optimize products and mold design as well as avoid errors in the products themselves [10,11,12,13]. The optimal gate position was selected and analyses were performed (part of the analyses performed is shown in Figure 2) that allow predicting surface defects and the quality of the mold surface, the possibility of shrinkage and air traps, the cooling time of the mold (and cooling to the shape retention temperature), the probability of filling the mold cavity, injection pressure, as well as the pressure drop in the mold cavity.

Through simulation, for the given parameters *mold temperature* 50°C and *melt temperature* 220°C, the optimal gate location was selected (Figure 2a), the predicted mold fill time is 0.3723 sec (Figure 2b) and the injection pressure is 0.914 MPa (Figure 2c).



a) Best gate location

b) Fill time

c) Injection pressure

d) Sink marks

e) Frozen layer fraction at end of fill

f) Cooling Time Variance

g) Volumetric shrinkage

h) Deflection

Figure 2: Polymer flow simulation in Moldflow Adviser software

In order to fill the mold for the selected PP material, a pressure of less than 1 MPa is required, which indicates that the required pressure can be achieved by hand force. Figure 2c) shows that on one side the injection pressure will be increased while on the other it is significantly lower. It should be noted that on one side there is a runner for leakage of excess melt. Injection pressure helps to fill all parts of the mold evenly. If the pressure is too low, the mold may not be completely filled, which can result in unfilled parts or defects. Injection pressure also affects the speed at which the material is injected into the mold. Higher pressure usually means faster injection speed.

Sink marks (Figure 2d) are the appearance of depressions on the surface of plastic parts, which are caused by uneven cooling and solidification of the material. Sink marks most often appear in places where the wall thickness is greater or near ribs and protrusions inside the mold. This phenomenon can affect the appearance and functionality of the final product.

Frozen layer fraction at end of fill (Figure 2e) analysis shows how much material has solidified (frozen) on the mold walls by the time the mold cavity is completely filled with molten material. Frozen layer fraction occurs in the middle of the injection molding process when the mold is fully filled, approximately 8%. Frozen layer fraction at end of fill depends on several factors such as mold temperature, molten material temperature, and injection speed. The frozen layer is formed due to contact of molten material with cooler mold surfaces, which causes rapid cooling and solidification of the material in contact with the mold. A high frozen layer fraction can cause defects such as poor layer bonding, increased internal stress, and poor mechanical properties of the product. If the material froze too quickly, it can make it difficult to properly fill the mold and result in inadequate filling or the defects appearance.

Cooling Time Variance (Figure 2f) represents the difference in time required to cool different parts of the plastic part to a certain temperature or to the moment when the part can be safely removed from the mold. High cooling time variance can lead to uneven shrinkage, deformation and surface quality problems.

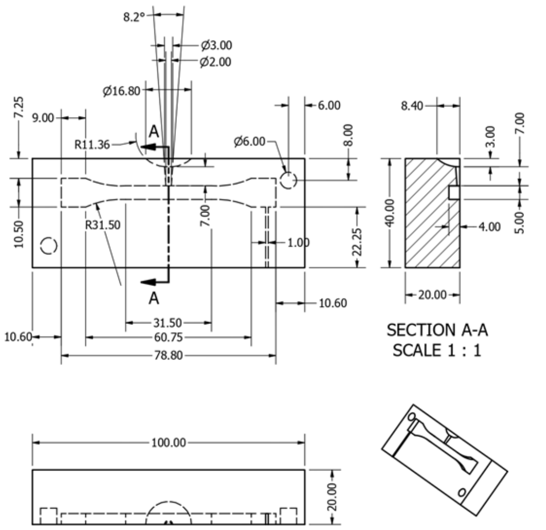
Volumetric shrinkage at ejection (Figure 2g) is a measure that shows the percentage of volume of a plastic part that has decreased due to cooling and solidification from the moment the mold is filled with molten material to the moment the part is removed from the mold. Volumetric shrinkage affects the final dimensions of the plastic part. Accurate shrinkage prediction allows mold designers to compensate for these changes to achieve accurate finished product dimensions. Uneven shrinkage can cause deformations, undulations or depressions on the surface of the part, which can affect the aesthetic and functional quality of the product.

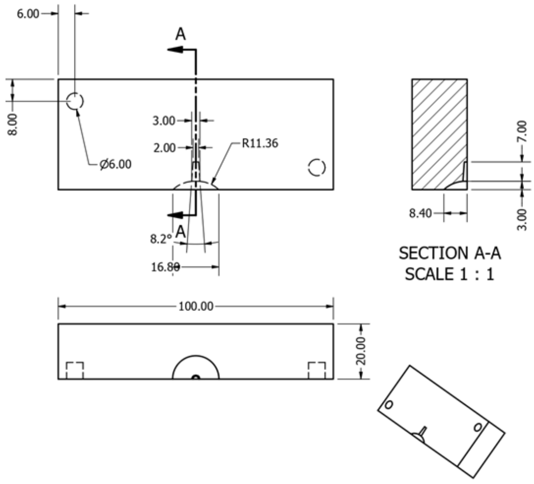
Deflection represents the visualization of deflection or displacement of plastic parts after the injection molding process, i.e. material bending. Figure 2h shows the distortion of the material (PP) in some parts of more than a millimeter. Deflection analysis shows how and to what extent a part deforms under the influence of internal stresses, temperature differences and uneven cooling. Material warping analysis is an analysis that simulates material shear in all axes (x,y,z) and how this affects the material.

III. EXPERIMENTAL PART

3.1 Design and manufacture of mold for injection molding

The two-part mold used in this experiment is shown in Figure 3. The mold was made of EN AW-6082 aluminum due to its weight reduction, ease of processing, and good thermal conductivity, which allows faster cooling of the product. The mold was designed using Autodesk Inventor 3D modeling software, which allows for the creation of precise models of the parts and molds. Mold dimensioning involves precisely defining the dimensions of the product according to specifications, ensuring uniform wall thickness for proper cooling and preventing deformation. Mold production involves processing the material using CNC machines for precise cutting and shaping of the mold. Finishing process includes polishing to ensure smooth surfaces that prevent parts from sticking, and surface treatment that may include coating to increase wear and corrosion resistance. Regular mold maintenance includes cleaning and lubricating the parts to ensure long-term functionality. Aluminum molds for plastic injection molding are essential for the production of precise and high-quality plastic parts, especially in the prototyping and small-scale production stages. Proper design, dimensioning and maintenance of these molds ensure high efficiency and quality of final products.





a) First half of the designed mold b) Second half of the designedmold

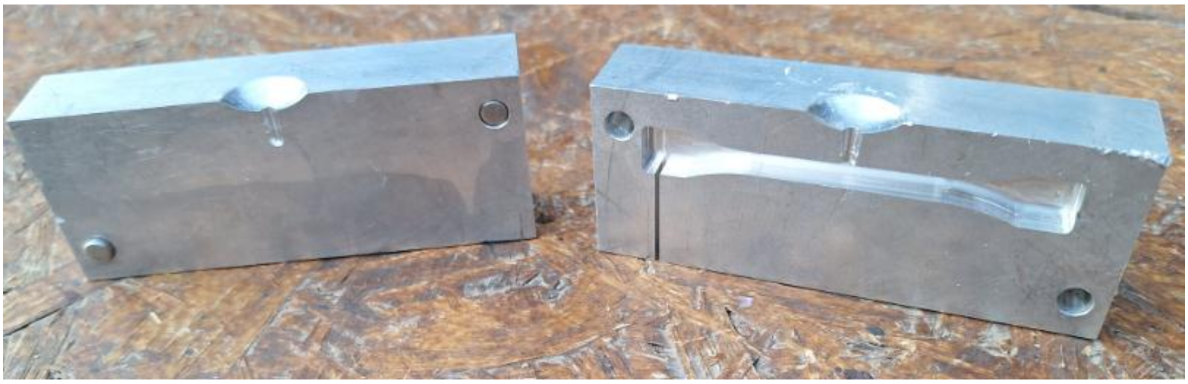


Figure 3: Aluminium mold for plastic injection molding

3.2 Design and manufacture of injection molding device

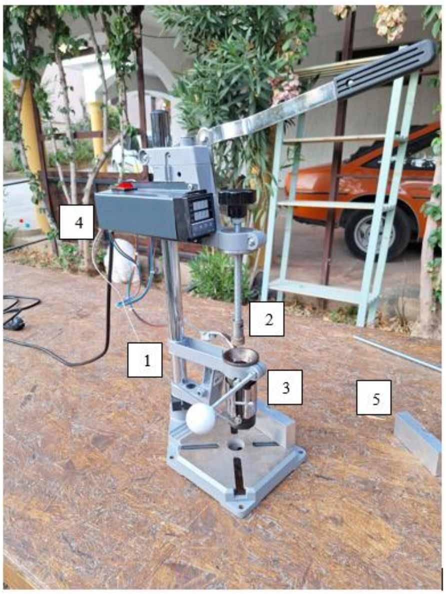
The following paper presents the design, assembly and testing of a customized plastic injection molding machine [14], intended for the production of test specimens made of polypropylene (PP). The machine uses a drill stand that allows pressure to be applied to the piston, which allows the injection of molten PP into the mold.

The injection molding machine consists of the following main components, indicated by numbers in Figure 4 b):

1. *Drill stand* - the main frame of the machine that provides stability and the necessary torque for piston pressure.
2. *Piston* - a manually operated lever that pushes the molten PP through the injection chamber and into the mold.
3. *Injection chamber* - a heated chamber in which PP is melted. The chamber is equipped with a digital temperature controller that maintains the required temperature for melting PP.
4. *Heater and digital controller* - an electric heater wrapped around the injection chamber melts PP, and the temperature is monitored and regulated via a digital controller.
5. *Mold* - a custom mold into which molten PP is injected to shape test specimens.

Individual parts of the In-house device for injection molding polymer materials are shown in Figure 5.

a) Drill stand



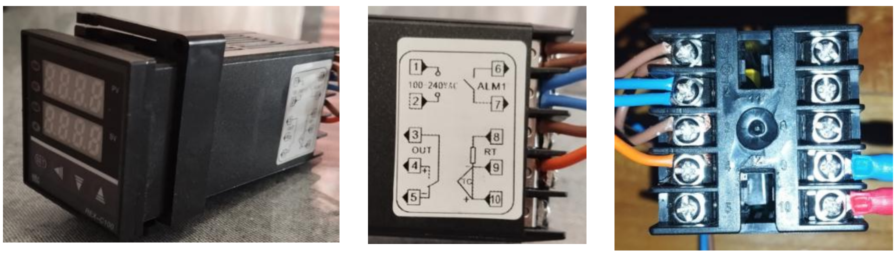
b) Parts of the injection molding device

Figure 4: Injection molding device

a) Piston centering plug

b) Chamber for polymer material

c) Assembly injection molding machine



d) Digital PID temperature controller REX-C100

f) Tubular heater

Figure 5: Parts of the In-house device for injection molding polymer materials

3.3 Operating principle of the device

Material granules, in this case polypropylene granules, are placed in the injection chamber. The heater is activated, melting the PP to a certain temperature, recommended in the range of 200-230°C. Once the PP is completely melted, the operator uses a lever to lower the plunger, which pushes the molten PP through the chamber into the mold. After injection, the mold is cooled, which hardens the PP into the desired shape. In the plastic injection molding process, precise temperature control is key to achieving optimal results. The temperature affects the viscosity of the polymer, the speed of filling the mold, the quality of the final processing of the product and, ultimately, the mechanical properties of the finished parts.

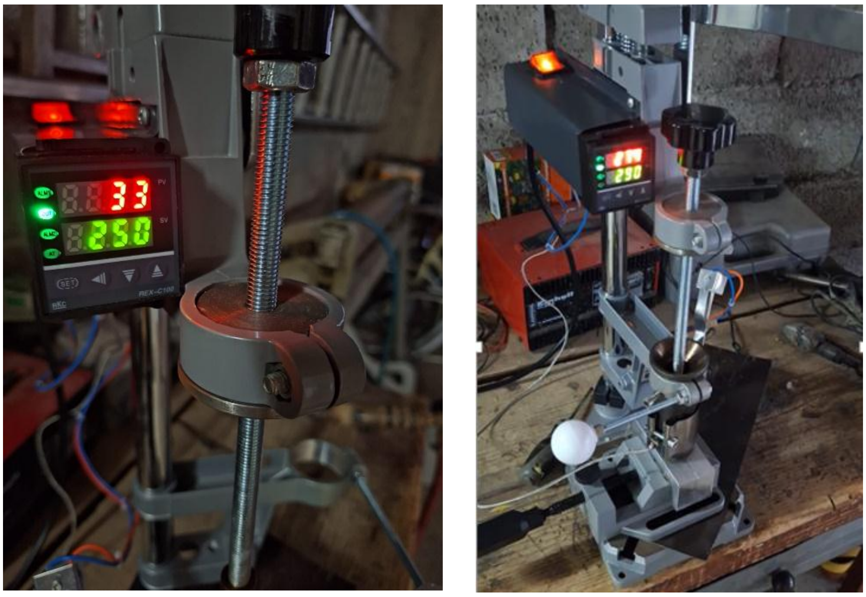
The digital temperature controller REX-C100 (Figure 5d) is used to control the temperature in the polypropylene (PP) melting chamber (Figure 5b). The regulator has two main indicators:

- red screen (PV - Process Value) that shows the current temperature in the chamber, read by the temperature sensor.
- green screen (SV - Set Value) showing the desired, target temperature set by the operator.

IV. RESULTS

4.1 Testing devices and measuring test specimens

Figure 6 shows a plastic injection molding machine in operation. Using a handle and threaded rod, the operator can apply the force required to inject the molten material into the mold. During operation, the machine maintains a set temperature to ensure consistency in the melting and injection process. When the material reaches the optimum viscosity, mechanical thrust allows it to be injected into the mold. Precise temperature maintenance is essential to achieve the desired properties of the test specimens, as any deviation can affect the quality and uniformity of the product. During operation, the machine maintains a set temperature to ensure consistency in the melting and injection process.



a) Parameters settings b) Device during injection process

Figure 6: Plastic injection molding device during operation

Figure 7a) shows a polypropylene test specimen cast in an aluminum mold. After the material has been injected into the mold and cooled to room temperature, the specimen must be removed from the mold without damaging it. Several tests on the test samples were conducted, as can be seen in Figure 7b), 7c) and 7d). Figure 7b) shows significant damage and deformation of the samples, which are the result of the technological processing, i.e. caused by entering inappropriate parameters and plastic injection into a previously unheated mold. During the injection molding process, the device was set to a temperature higher than recommended.

a) Two-part aluminium mold | b) First test samples series | c) Second test samples series | d) Third test samples series

Figure 7: Test specimens

Figure 7c) shows samples with a much more regular shape, but still contain excess polymer material on the edges. This excess, also known as "overflow," is created during the injection molding or casting process. Overflow is an inevitable result of the manufacturing process when plastic passes through the mold and in some cases, if the mold is slightly misaligned or if the plastic pressure is excessive, material leaks out. These overhangs are not only an aesthetic problem, but can also affect the mechanical properties of the specimen. The presence of overhangs can lead to inaccurate results in mechanical tests such as tensile strength, where irregularities will cause stress concentration and potential premature failure of the specimen. Therefore, it is important to remove all overhangs after forming the specimens to ensure the accuracy and repeatability of the test results.

Figure 7d) shows the test samples from the third series of testing. Each sample was made at different process parameters, i.e. different temperatures. In the upper samples (according to Figure 7d), a yellow hue prevails, indicating unfavorable process parameters, while the last two samples are of much higher quality (clean transparent sample). We can conclude that due to the too high temperature during the injection of polypropylene, the material properties were degraded, which significantly impaired both the visual and mechanical properties of the test samples.

The test specimens will be further used for testing the mechanical properties (tensile strength) on a tensile test machine [15], which is also made as a part of the master thesis at the University Department of Professional Studies.

V. CONCLUSION

A very important role in the injection molding process is played by correctly set process parameters, especially the temperature of both the mold and the material. A heater with a digital controller allows for precise regulation of the material temperature, which is a key factor in achieving an optimal plastic melting and injection process. Injection parameters, such as temperature and pressure, can be easily controlled and adjusted, ensuring high process repeatability and sample quality.

Using a drill stand as the basic piston pushing mechanism has proven to be a practical, effective and economical solution for making small batches of samples from polymer materials. Based on the additional experiments conducted and the results obtained, it can be concluded that the manufactured plastic injection molding device is functional and reliable and has met the set requirements. The device will continue to be used in the laboratory of the University Department of Professional Studies for testing and analyzing the mechanical properties of polymer materials and will be upgraded with a temperature controller that will be placed on the mold to maintain the recommended temperature.

REFERENCES

[1]. Silin, C. Si, Guimao, S. 2022. Design of an Automobile Injection Mould Based on Automation Technology. Mobile Information Systems. 2022. 1-13. 10.1155/2022/8224364.

[2]. Huiwen, M., Youmin, W., Deyu, Y. 2021. Study of Injection Molding Process Simulation and Mold Design of Automotive Back Door Panel. Journal of Mechanical Science and Technology. No.5, pp. 2331-2344. https://doi.org/10.1007/s12206-022-0415-0.

[3]. Ankush Mourya, Arshit Nanda, Kartik Parashar, Sushant, Rajender Kumar. 2023. An explanatory study on defects in plastic molding parts caused by machine parameters in injection molding process. *Materials Today: Proceedings*, 78 (3), pp 656-661. ISSN 2214-7853, https://doi.org/10.1016/j.matpr.2022.12.070.

[4]. Farotti, E., Natalini, M. 2018. Injection molding. Influence of process parameters on mechanical properties of polypropylene polymer. A first study.,Procedia Structural Integrity, 8, pp 256-264, ISSN2452-3216, https://doi.org/10.1016/j.prostr.2017.12.027.

[5]. Pezer, D. (2020). Plastic Products Design: Today's Challenges from the Aspect of Environmental Protection. 4th International Scientific and Professional Conference Contemporary Issues in Economy and Technology (CIET), Split, pp 453-461.

[6]. Simonazzi, T., Haylock, J.C. 1995. Recent Advances in Polypropylene-Based Materials, Properties and Applications. In: Prasad, P.N., Mark, J.E., Fai, T.J. (eds) Polymers and Other Advanced Materials. Springer, Boston, MA. https://doi.org/10.1007/978-1-4899-0502-4_20.

[7]. Hossain MT, Shahid MA, Mahmud N, Habib A, Rana MM, Khan SA, Hossain MD. 2024. Research and application of polypropylene: a review. Discov Nano. 19(1):2. doi: 10.1186/s11671-023-03952-z.

[8]. "British Plastics Federation." Polypropylene (PP). N.p., n.d. Web. 15 Sep. 2024. <http://www.bpf.co.uk/plastipedia/polymers/pp.aspx>

[9]. ISO Standard: ISO 527-2:2012 Plastics — Determination of tensile properties — Part 2: Test conditions for moulding and extrusion plastics.

[10]. Azuddin, M., Zahari, T., Imtiaz, C. (2011). Observation of Polypropylene (PP) Melt Flow on Macro and Micro Cavities during Filling Phase of Injection Molding. Advanced Materials Research. 314-316. 1273-1277. 10.4028/www.scientific.net/AMR.314-316.1273.

[11]. Mathieu, L., Stockmann, L., Haudin, J-M., Monasse, B.,Vincent, M., Barthez, J.-M., C. et al. (2001). Flow Marks in Injection Molding of PP: Influence of Processing Conditions and Formation in Fountain Flow. International Polymer Processing Journal of the Polymer Processing Society. 16. 404-411. 10.1515/ipp-2001-0012.

[12]. Jain, K., Somwanshi, D., Jain, A. 2021. Effect of Process Parameter on Plastic Parts Using ANOVA with Moldflow Simulation. In: Patnaik, A., Kozeschnik, E., Kukshal, V. (eds) Advances in Materials Processing and Manufacturing Applications. iCADMA 2020. Lecture Notes in Mechanical Engineering. Springer, Singapore. https://doi.org/10.1007/978-981-16-0909-1_43.

[13]. He, H. B., Wu, F. L., Deng, Y. M. 2011. Injection Molding Process Parameter Optimization for Warpage Minimization Based on Moldflow. In Applied Mechanics and Materials. 101–102, pp. 254–257). Trans Tech Publications, Ltd. https://doi.org/10.4028/www.scientific.net/amm.101-102.254.

[14]. Atelj, J. Device for injection molding of polymer test samples. University department of Professional Studies, University of Split, Split, Croatia, Master thesis, 2024.

[15]. Pezer, D., Vukas, F., Butir, M. 2022. Experimental Study of Tensile Strength for 3D Printed Specimens of HI-PLA Polymer Material on In-house Tensile Test Machine. *Technium: Romanian Journal of Applied Sciences and Technology,* 4, 10; 197-206 doi:10.47577/technium.v4i10.7927.